Electro-Thermo-Mechanical Simulations of Aluminum Bond Wires in IGBT-Packages

C. Hager, Y. Tronel and W. Fichtner

Swiss Federal Institute of Technology Zurich
Integrated Systems Laboratory
Gloriastrasse 35, CH–8092 Zurich, Switzerland
hager@iis.ee.ethz.ch, tronel@iis.ee.ethz.ch, fw@iis.ee.ethz.ch

ABSTRACT

Finite-element simulations have been performed on aluminum wires as used in modern IGBT (Insulated Gate Bipolar Transistor) power modules. These wires were ultrasonically bonded to thin molybdenum plates and the influence of different origins of the mechanical stresses on the bond interface has been studied. The stress origins considered were: 1. heating and expansion of the bond wire due to an electric current, 2. the thermal expansion of a heated base plate and 3. the heating of the interface of the two materials with different thermal expansion coefficients. The geometries under investigation were: a) Wedge bonds on both ends and b) a wedge bond on one end and a ball bond on the other end.

It was found that the stresses at the interface are dominated by the different thermal expansion coefficients of the aluminum bond wire and the molybdenum bond pad material. These main stresses at the interface are localized within a layer of about 50 μm and are generally lower for the ball bond connection than for the wedge bond geometry.

Keywords: Wire bonding, electro-thermo-mechanical simulations, finite element simulations

INTRODUCTION

Bond wire lift-off is one of the major failure types in modern IGBT modules used for current switching in high power traction applications (locomotives, trams, etc.) [1][2]. During operation the modules undergo thermal cycling, leading to fatigue and cracking of the pure aluminum wire (99.99 %), which is easily bonded to the dye but extremely soft. Especially the region of the wire near the bond interface is the location for plastic deformations and fatigue cracking. Because the current connections are usually made by a set of parallel bond wires, the rupture of one wire leads in an increased load on the remaining. This makes it a self-accelerating process, ultimately leading to the complete failure of the single device and the entire module. Therefore, to increase the lifetime of IGBT modules, it is extremely important to increase the reliability of the wire bond connections.

The most widely accepted methods for controlling the quality of wire bonding and estimating the reliability is the wire bond pull test, often in combination with thermal cycling tests. These tests are extremely time consuming and expensive, especially if lifetimes of a few decades have to be estimated. It is therefore desired to have a faster and cheaper method during the design and optimization process, which allows to find a way to obtain the specified reliability and lifetime.

Using appropriate software tools can considerably speed up the design process and additionally cut costs, if unnecessary experimental processes can be replaced by cheaper and more flexible computer simulations.

As a first step in an optimization process of wire bond connections in IGBT modules, linear electro-thermo-mechanical simulations have been performed to reveal the stress levels and stress distributions as well as the contributions of different stress sources. In particular the stresses at the interface between the bond wire and the molybdenum plate, which acts as the bond pad, were studied. Simulations were performed using the finite element tool SOLIDIS$_{\text{-ISE}}$ [3][4], which is tailored to the coupled simulation of electro-thermo-mechanical phenomena. It was not intended to extract lifetime information from this simulations, but to evaluate the relative influence of the different phenomena on the mechanical stresses at the bond wire – bond pad interface. For more realistic simulations, phenomena like plasticity, creep and fatigue cracking have to be taken into account as well as the complicated material properties at the bond interface. Including these phenomena results in a considerably greater computational effort. In the beginning of a design process, where a lot of parameters are undetermined, it is necessary to get informations about the importance of the different variables. This is therefore the phase where linear simulations are employed. Subsequently, the model extracted from these first calculations can be used in more detailed studies.

The purpose of this work is to present a first study of mechanical stresses at the bond interface, resulting from two different bonding geometries and to reveal the contributions and importance of different stress origins. Geometries under investigation were: a) Wedge bonds on both ends and b) a wedge bond on one end and a ball bond on the other end. The stress sources considered

were: 1. heating and expansion of the bond wire due to an electric current, 2. the expansion of the base plate and 3. the heating of the interface of the two materials with different thermal expansion coefficients.

SIMULATION GEOMETRIES AND MESH GENERATION

The setup of the simulation geometries followed bonding geometries of real IGBT modules as provided by ABB Corporate Research Ltd. (Baden-Dättwil, Switzerland). For building the initial finite element mesh, the tensor grid editor TED$_{\text{-ISE}}$ [5] was used. Figures 1 to 4 show the initial finite element meshes with detailed views of the bond areas.

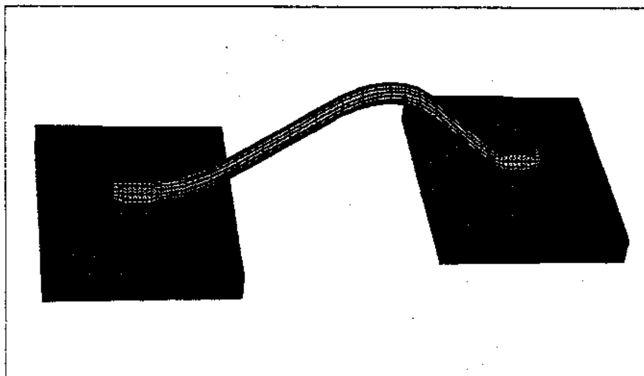

Figure 1: Aluminum wire, wedge-wedge bonded on molybdenum

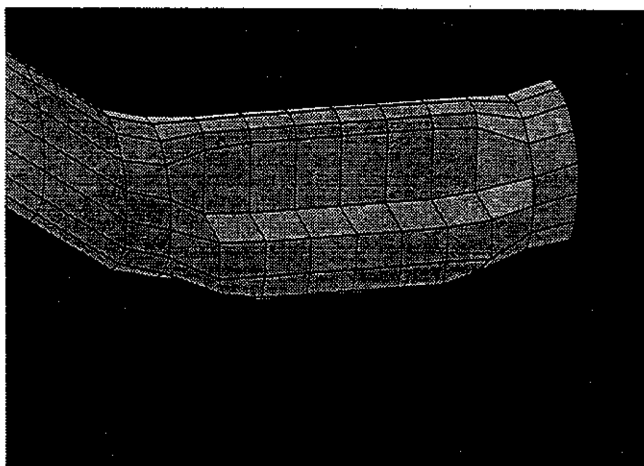

Figure 2: Wedge bond

In a typical traction module the IGBTs are soldered on a DCB (Direct Copper Bonded) substrate. The Al-wires are either directly bonded to the top of the Dye or a strain buffer is soldered on top of the Dye and the Al-wires are bonded to this buffer layer to increase the reliability of the bond wire connections. In this paper only the second possibility is investigated. The strain buffer was assumed to be a thin molybdenum plate. The DCB substrate itself is soldered onto the thick copper base plate. For these simulations the dye, DCB substrate and base plate structure were not modeled because in global simulations, the areas of interest are the bond wire and the bond wire – bond pad interface. Thus, the behavior of not modeled parts had to be represented in the boundary conditions and were approximated as copper, because the thick copper base plate dominates its mechanical behavior.

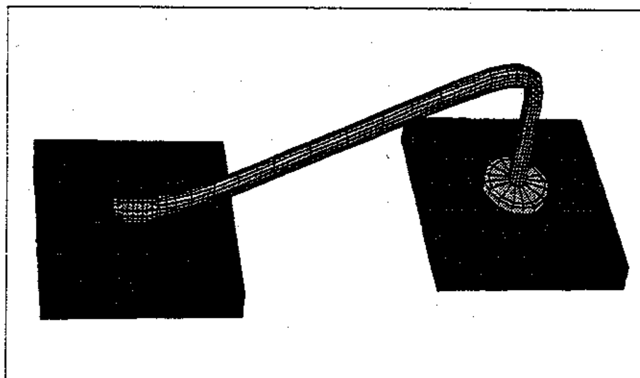

Figure 3: Aluminum wire, wedge-ball bonded on molybdenum

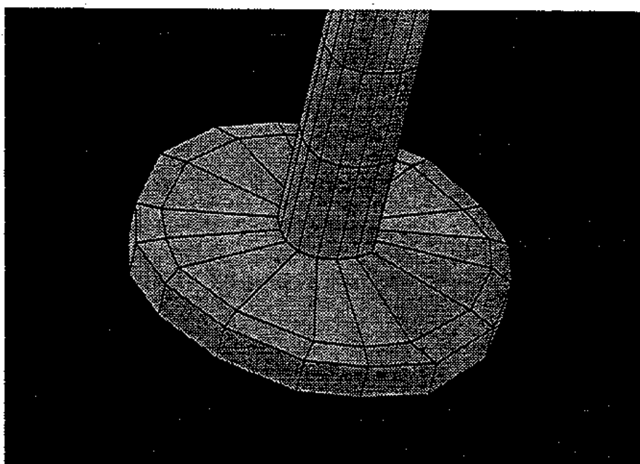

Figure 4: Ball bond

As wire material pure aluminum (99.99 %) was used and the wire diameter was 0.350 mm. The distance between the centers of the bonds was 8.8 mm and the loop heights were 2.3 mm and 3 mm for the wedge-wedge and wedge-ball geometry, respectively. The level of the right bond was 0.750 mm above the left bond.

SIMULATION AND BOUNDARY CONDITIONS

The simulation conditions were chosen to represent a typical state of operation of the system. This results in an electric direct current (DC) through the wire of 3 A, a temperature of the copper base plate of 50 °C and 100 °C at the molybdenum plate. The stress free temperature was chosen as 25 °C.

Heating of the bond wire with an electric current

The bond wire was heated by a DC of 3 A. Thus, a complete electro-thermo-mechanical simulation was performed. The molybdenum plate displacement was prescribed as zero.

Expansion of the base plate

The amount of expansion of the copper base plate was not simulated, but estimated by the following simple procedure and used as boundary condition for the molybdenum plate in the simulation. The thermal expansion coefficient α for copper is 17 ppm/K. Using the simple formula

$$l = l_0 \cdot (1 + \alpha \cdot \Delta T) \tag{1}$$

for the thermal expansion, one obtains $l - l_0 = 3.75\,\mu\text{m}$ for $l_0 = 8.8\,\text{mm}$ and $\Delta T = 25\,\text{K}$. For the boundary conditions the left molybdenum plate was kept fixed and for the right plate a displacement of $\Delta x = 3.75\,\mu\text{m}$ was prescribed, with the +x-direction starting at the left bond and heading towards the right bond.

Stress due to different thermal expansion coefficients

To evaluate the stress induced by different thermal expansion coefficients, a heat source was placed at the bottom planes of the molybdenum plates, heating the entire structure. The prescribed temperature was $100\,°C$ ($\Delta T = 75\,\text{K}$). The molybdenum plates were allowed to expand freely, to extract only the stresses due to the interface heating.

MESH REFINEMENT AND CONVERGENCE BEHAVIOR

In order to obtain a sufficient accuracy, the solution obtained using the relatively coarse initial mesh was used to compute a refinement indicator. Such a function is computed element-wise, which allows for a mesh refinement according to the local solution. In this way, a near-optimal mesh results, containing just enough elements to obtain the desired accuracy. These elements are thus introduced only where they are needed in order to obtain the required solution.

Within these simulations, the so-called mixed mode indicator ψ_{MM} was used as the refinement indicator [6]. This function arises when the linear and constant parts (these can be exactly solved for by the FE method employed) are subtracted from the solution. As the constant term does not contribute to the energy of the system, the definition of the mixed-mode refinement function is

$$\psi_{MM} = W\left(1 - \frac{\psi_{linear}}{\psi - \psi_{const}}\right) \tag{2}$$

Here, the mechanical energy W represented by an element can be expressed in terms of the strain (ϵ) and stress (σ) values:

$$W = \frac{1}{2}\int_{\Omega} \epsilon^T \sigma d\Omega \tag{3}$$

Elements with a mixed mode value larger than the average value of the entire element ensemble have been refined. Refinement of an element means the division of the element along all axes, i.e. a refined 3D-element gives rise to eight elements in the next solution step. This step can be repeated, leading to a recursive octree-mesh.

The number of elements required was determined by introducing a convergence criterion based on the von Mises stress. Here, convergence of the von Mises stress near the singular point in Fig. 6 was taken as definition of overall convergence. As an example, the influence of the number of elements used in the ball bond interface heating simulation on the convergence criterion is shown in Fig. 5. After at least five refinement steps, the solution at the point under investigation could be considered as converged. Absolute von Mises stress values given in this paper are determined by checking the convergence due to mesh refinements as described above.

RESULTS AND DISCUSSION

The main contribution to the stress at the bond wire – bond pad interface is due to the different thermal expansion coefficients of aluminum and molybdenum. This effect dominates the others by more than one order of magnitude. For both geometries the mechanical stress peaks at the edges of the interface where the aluminum wire leaves the molybdenum plate, especially in the middle of the structure (the plane of symmetry). Figure 6 shows the initial geometry of the ball bond near the interface and the refined finite element mesh after five adaptive refinement steps, that were necessary to achieve convergence in the region $5\,\mu\text{m}$ from the singular point. The location of high stresses and the fact that they exceed the yield stress of aluminum by about

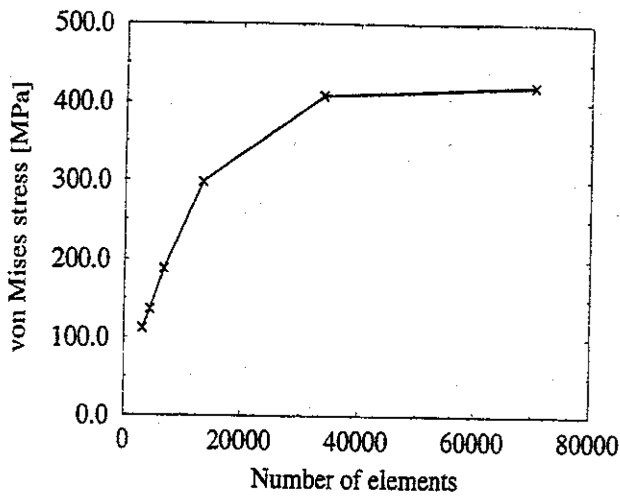

Figure 5: Example of the convergence check as performed for the ball bond - bond pad interface heating simulation. Evolution of the von Mises stress value near the singular point shown in Fig. 6, with increasing number of elements introduced by adaptive refinement

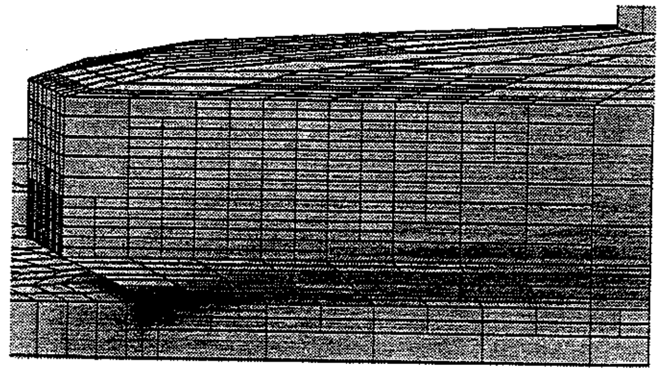

Figure 6: Cut through the middle of the ball bond with the initial geometry and refined mesh near the interface, showing the von Mises stress distribution. Light grey indicates low stress values, while dark grey represents high von Mises stresses peaking at the interface, arising especially at the edge where the ball bond leaves the molybdenum plate

a factor of 10 is consistent with the observed fatigue pattern in this region, because plastic deformations give rise to the propagation of micro cracks that were initialized during the bonding process [7]. It is important to recognize that the stresses for the ball bond geometry are only half as high as for the wedge bond geometry. A possible explanation might be the different bond foot area, which is 1.5 times larger for the ball bond.

The influence of the other two stress sources on the interface stresses is much smaller. Especially the design of the ball bond shifts the maximum stresses originating from these sources to the region above the deformed ball, yielding in much lower stresses at the interface compared to the wedge bond. The maximal stresses induced at the wedge bond interface are located like the stresses due to the different expansion coefficients. In addition to the stresses at the interface, high stresses appear at the bond heel (see Fig. 7).

The results of the different simulations are summarized in Table 1.

It is important to keep in mind that the results given are achieved by employing only elasticity and no plasticity or creep effects. That means that stresses exceeding the elastic limit of aluminum (approximately 25 MPa) are overestimated. Nevertheless, the comparison of the results of the different geometries reveals which geometry will produce more plastic deformations and therefore will be more exposed to micro crack growth and fatigue.

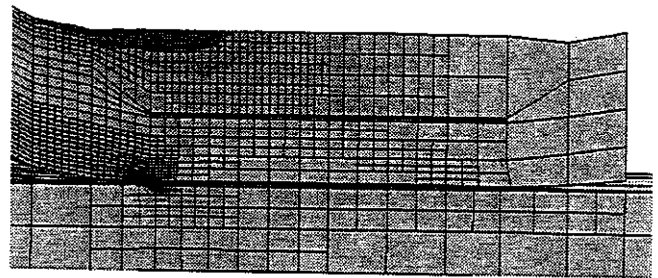

Figure 7: Cut through the middle of the wedge bond with the initial geometry and refined mesh showing the von Mises stress distribution. Light grey indicates low stress values, while dark grey represents high von Mises stresses peaking especially at the edge where the wedge bond leaves the molybdenum plate and at the bond heel

Table 1: Simulation Results (von Mises stress) [Pa]

	wedge bond	ball bond
wire heating	$5.0 \cdot 10^6$	$5.0 \cdot 10^4$
base plate expansion	$2.6 \cdot 10^7$	$1.2 \cdot 10^5$
interface heating	$7.8 \cdot 10^8$	$4.1 \cdot 10^8$

CONCLUSIONS

Global finite element simulations on two common bond wire geometries were performed, employing linear electro-thermo-mechanical calculations. The contributions of different stress sources to the stresses at the bond interface were investigated. The stress at the bond wire – bond pad interface is dominated by the stress due to the different thermal expansion coefficients. The contributions originating from base plate expansion and heating of the bond wire due to an electric current are at least one order of magnitude smaller. The influence of the bonding geometry were studied by comparing the results for a wedge-wedge and a wedge-ball geometry. For the ball bond, the stress at the interface is generally

lower than for the wedge bond configurations. These elastic calculations have shown stress levels that exceed the yield stress of aluminum, which is consistent with the observation of plastic deformations and micro crack growth the interface.

ACKNOWLEDGMENTS

The authors would like to thank Dr. Alexander Stuck (ABB Corporate Research Ltd., Baden-Dättwil, Switzerland) for providing the bonding geometries and material data. Special thanks go to Dr. Hans-Petter Lien and Dr. Lars Bomholt (ISE AG, Zurich, Switzerland) for all the fruitful discussions and Mr. Georg Hertkorn for his extensive work on TED_{-ISE}. Support for this work is provided by a grant from the Swiss Government within the project BONDOPT (project number 4.07) as part of the swiss priority program MINAST.

REFERENCES

[1] M.H. Poech, K.J. Dittmer and D. Gäbisch, "Investigations on the Damage Mechanism of Aluminium Wire Bonds used for High-Power Applications", EuPac '96

[2] K.J. Dittmer, M.H. Poech, F.W. Wulff and M. Krumm, "Failure analysis of Aluminum wire bonds in high power IGBT Modules", Mat.Res.Soc.Symp.Proc.Vol. 390, 1995, pp. 251-256

[3] Jörg M. Funk, J. G. Korvink, Johannes Bühler, Martin Bächtold, Henry Baltes, "SOLIDIS: A Tool for Microactuator Simulation in 3-D", Journal of Micromechanical Systems, Vol. 6, No. 1, March 1997, pp. 70-82

[4] Jörg M. Funk, Lars H. Bomholt, Renzo P. Paganini, Hans-Petter Lien, Wolfgang Fichtner, "SOLIDIS: A TCAD Environment for Packaging Simulation", EEP-Vol. 19-1, Advances in Electronic Packaging - 1997, Volume 1, pages 919-926, ASME 1997

[5] ISE Integrated Systems Engineering AG, "TED_{-ISE} user manual", ISE TCAD Release 5.0 Manual Part 27, ISE AG (1998)

[6] ISE Integrated Systems Engineering AG, "$SOLIDIS_{-ISE}$ user manual", ISE TCAD Release 5.0 Manual Part 26, ISE AG (1998)

[7] J.F. Haag, A. Kolbeck "Stand und Entwicklung der Drahtbondtechnik", Frauenhofer-Institut für Festkörpertechnologie, München 1990